Abaqus 6.12

Abaqus Interface for Moldflow User's Manual

Abaqus Interface for Moldflow

User's Manual

Legal Notices

CAUTION: This documentation is intended for qualified users who will exercise sound engineering judgment and expertise in the use of the Abaqus Software. The Abaqus Software is inherently complex, and the examples and procedures in this documentation are not intended to be exhaustive or to apply to any particular situation. Users are cautioned to satisfy themselves as to the accuracy and results of their analyses.

Dassault Systèmes and its subsidiaries, including Dassault Systèmes Simulia Corp., shall not be responsible for the accuracy or usefulness of any analysis performed using the Abaqus Software or the procedures, examples, or explanations in this documentation. Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

The Abaqus Software is available only under license from Dassault Systèmes or its subsidiary and may be used or reproduced only in accordance with the terms of such license. This documentation is subject to the terms and conditions of either the software license agreement signed by the parties, or, absent such an agreement, the then current software license agreement to which the documentation relates.

This documentation and the software described in this documentation are subject to change without prior notice.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiary.

The Abaqus Software is a product of Dassault Systèmes Simulia Corp., Providence, RI, USA.

© Dassault Systèmes, 2012

Abaqus, the 3DS logo, SIMULIA, CATIA, and Unified FEA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the United States and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the Abaqus 6.12 Installation and Licensing Guide.

Locations

SIMULIA Worldwide Headquarters	Rising Sun Mills, 166 Valley Street, Providence, RI 02909–2499, Tel: +1 401 276 4400, Fax: +1 401 276 4408, simulia.support@3ds.com, http://www.simulia.com
SIMULIA European Headquarters	Stationsplein 8-K, 6221 BT Maastricht, The Netherlands, Tel: +31 43 7999 084, Fax: +31 43 7999 306, simulia.europe.info@3ds.com

Dassault Systèmes' Centers of Simulation Excellence

United States	Fremont, CA, Tel: +1 510 794 5891, simulia.west.support@3ds.com
	West Lafayette, IN, Tel: +1 765 497 1373, simulia.central.support@3ds.com
	Northville, MI, Tel: +1 248 349 4669, simulia.greatlakes.info@3ds.com
	Woodbury, MN, Tel: +1 612 424 9044, simulia.central.support@3ds.com
	Mayfield Heights, OH, Tel: +1 216 378 1070, simulia.erie.info@3ds.com
	Mason, OH, Tel: +1 513 275 1430, simulia.central.support@3ds.com
	Warwick, RI, Tel: +1 401 739 3637, simulia.east.support@3ds.com
	Lewisville, TX, Tel: +1 972 221 6500, simulia.south.info@3ds.com
Australia	Richmond VIC, Tel: +61 3 9421 2900, simulia.au.support@3ds.com
Austria	Vienna, Tel: +43 1 22 707 200, simulia.at.info@3ds.com
Benelux	Maarssen, The Netherlands, Tel: +31 346 585 710, simulia.benelux.support@3ds.com
Canada	Toronto, ON, Tel: +1 416 402 2219, simulia.greatlakes.info@3ds.com
China	Beijing, P. R. China, Tel: +8610 6536 2288, simulia.cn.support@3ds.com
	Shanghai, P. R. China, Tel: +8621 3856 8000, simulia.cn.support@3ds.com
Finland	Espoo, Tel: +358 40 902 2973, simulia.nordic.info@3ds.com
France	Velizy Villacoublay Cedex, Tel: +33 1 61 62 72 72, simulia.fr.support@3ds.com
Germany	Aachen, Tel: +49 241 474 01 0, simulia.de.info@3ds.com
	Munich, Tel: +49 89 543 48 77 0, simulia.de.info@3ds.com
India	Chennai, Tamil Nadu, Tel: +91 44 43443000, simulia.in.info@3ds.com
Italy	Lainate MI, Tel: +39 02 3343061, simulia.ity.info@3ds.com
Japan	Tokyo, Tel: +81 3 5442 6302, simulia.jp.support@3ds.com
	Osaka, Tel: +81 6 7730 2703, simulia.jp.support@3ds.com
Korea	Mapo-Gu, Seoul, Tel: +82 2 785 6707/8, simulia.kr.info@3ds.com
Latin America	Puerto Madero, Buenos Aires, Tel: +54 11 4312 8700, Horacio.Burbridge@3ds.com
Scandinavia	Stockholm, Sweden, Tel: +46 8 68430450, simulia.nordic.info@3ds.com
United Kingdom	Warrington, Tel: +44 1925 830900, simulia.uk.info@3ds.com

Authorized Support Centers

Argentina	SMARTtech Sudamerica SRL, Buenos Aires, Tel: +54 11 4717 2717
	KB Engineering, Buenos Aires, Tel: +54 11 4326 7542
	Solaer Ingeniería, Buenos Aires, Tel: +54 221 489 1738
Brazil	SMARTtech Mecânica, Sao Paulo-SP, Tel: +55 11 3168 3388
Czech & Slovak Republics	Synerma s. r. o., Psáry, Prague-West, Tel: +420 603 145 769, abaqus@synerma.cz
Greece	3 Dimensional Data Systems, Crete, Tel: +30 2821040012, support@3dds.gr
Israel	ADCOM, Givataim, Tel: +972 3 7325311, shmulik.keidar@adcomsim.co.il
Malaysia	WorleyParsons Services Sdn. Bhd., Kuala Lumpur, Tel: +603 2039 9000, abaqus.my@worleyparsons.com
Mexico	Kimeca.NET SA de CV, Mexico, Tel: +52 55 2459 2635
New Zealand	Matrix Applied Computing Ltd., Auckland, Tel: +64 9 623 1223, abaqus-tech@matrix.co.nz
Poland	BudSoft Sp. z o.o., Poznań, Tel: +48 61 8508 466, info@budsoft.com.pl
Russia, Belarus & Ukraine	TESIS Ltd., Moscow, Tel: +7 495 612 44 22, info@tesis.com.ru
Singapore	WorleyParsons Pte Ltd., Singapore, Tel: +65 6735 8444, abaqus.sg@worleyparsons.com
South Africa	Finite Element Analysis Services (Pty) Ltd., Parklands, Tel: +27 21 556 6462, feas@feas.co.za
Spain & Portugal	Principia Ingenieros Consultores, S.A., Madrid, Tel: +34 91 209 1482, simulia@principia.es

Taiwan	Simutech Solution Corporation, Taipei, R.O.C., Tel: +886 2 2507 9550, camilla@simutech.com.tw
Thailand	WorleyParsons Pte Ltd., Singapore, Tel: +65 6735 8444, abaqus.sg@worleyparsons.com
Turkey	A-Ztech Ltd., Istanbul, Tel: +90 216 361 8850, info@a-ztech.com.tr

Complete contact information is available at http://www.simulia.com/locations/locations.html.

Preface

This section lists various resources that are available for help with using Abaqus Unified FEA software.

Support

Both technical engineering support (for problems with creating a model or performing an analysis) and systems support (for installation, licensing, and hardware-related problems) for Abaqus are offered through a network of local support offices. Regional contact information is listed in the front of each Abaqus manual and is accessible from the **Locations** page at www.simulia.com.

Support for SIMULIA products

SIMULIA provides a knowledge database of answers and solutions to questions that we have answered, as well as guidelines on how to use Abaqus, SIMULIA Scenario Definition, Isight, and other SIMULIA products. You can also submit new requests for support. All support incidents are tracked. If you contact us by means outside the system to discuss an existing support problem and you know the incident or support request number, please mention it so that we can query the database to see what the latest action has been.

Many questions about Abaqus can also be answered by visiting the **Products** page and the **Support** page at www.simulia.com.

Anonymous ftp site

To facilitate data transfer with SIMULIA, an anonymous ftp account is available at **`ftp.simulia.com`**. Login as user **`anonymous`**, and type your e-mail address as your password. Contact support before placing files on the site.

Training

All offices and representatives offer regularly scheduled public training classes. The courses are offered in a traditional classroom form and via the Web. We also provide training seminars at customer sites. All training classes and seminars include workshops to provide as much practical experience with Abaqus as possible. For a schedule and descriptions of available classes, see www.simulia.com or call your local office or representative.

Feedback

We welcome any suggestions for improvements to Abaqus software, the support program, or documentation. We will ensure that any enhancement requests you make are considered for future releases. If you wish to make a suggestion about the service or products, refer to www.simulia.com. Complaints should be made by contacting your local office or through www.simulia.com by visiting the **Quality Assurance** section of the **Support** page.

Contents

1. Introduction

1.1 What information does this manual contain? . . . 1–1
1.2 What is the Abaqus Interface for Moldflow? . . . 1–1
1.3 What is the general procedure for using the Abaqus Interface for Moldflow? . . . 1–2

2. Description of the translator

2.1 The Moldflow simulation . . . 2–1
2.2 The Moldflow interface files . . . 2–1
2.3 Assumptions used to translate the Moldflow data for midplane simulations . . . 2–3
2.4 Assumptions used to translate the Moldflow data for three-dimensional solid simulations . . . 2–4
2.5 Files created by the Abaqus Interface for Moldflow . . . 2–5
2.5.1 Files created for a midplane simulation . . . 2–5
2.5.2 Files created for a three-dimensional solid simulation . . . 2–6

3. Using the Abaqus Interface for Moldflow

3.1 Execution procedure for the Abaqus Interface for Moldflow . . . 3–1
3.2 Preparing the Abaqus input (`.inp`) file for analysis . . . 3–2
3.2.1 Preparing for a shrinkage and warpage analysis . . . 3–2
3.2.2 Preparing for a service loading analysis . . . 3–3
3.2.3 Preparing for other analysis types . . . 3–3

4. Examples

4.1 Extracting example problem files . . . 4–1
4.2 Example 1: Natural frequency analysis of a fiber-filled bracket . . . 4–1
4.3 Example 2: Natural frequency analysis of an unfilled bracket . . . 4–5
4.4 Example 3: Deformation due to initial stresses in a three-dimensional filled bracket . . . 4–7

1. Introduction

This chapter provides an overview of the Abaqus Interface for Moldflow. The following topics are covered:

- "What information does this manual contain?," Section 1.1
- "What is the Abaqus Interface for Moldflow?," Section 1.2
- "What is the general procedure for using the Abaqus Interface for Moldflow?," Section 1.3

The installation of the Abaqus Interface for Moldflow is included in the Abaqus product installation. For information on installing Abaqus, see the Abaqus Installation and Licensing Guide.

1.1 What information does this manual contain?

This manual explains how to use the Abaqus Interface for Moldflow for midplane (three-dimensional shell) and three-dimensional solid simulations. For detailed information about using Moldflow, see the Moldflow documentation collection available from Autodesk, Inc.

1.2 What is the Abaqus Interface for Moldflow?

Autodesk, Inc. creates simulation software that is used by the plastics injection molding industry. One of these programs, Moldflow Plastics Insight (referred to as Moldflow in this manual), models the mold-filling process. The results of a Moldflow simulation include calculations of material properties and residual stresses in the plastic part.

The Abaqus Interface for Moldflow translates finite element model information from a Moldflow analysis into a partial Abaqus input file. The translator requires the Moldflow interface files that are created by the Moldflow analysis. (See "The Moldflow interface files," Section 2.2, for more information.)

For midplane simulations, the Abaqus Interface for Moldflow reads the interface (**`.pat`** and **`.osp`**) files created by Abaqus Interface for Moldflow Version MPI 3 or later.

For three-dimensional solid simulations using Moldflow Version MPI 6, the Abaqus Interface for Moldflow reads the interface (**`.inp`** and **`.xml`**) files created using the Visual Basic script **`mpi2abq.vbs`**. This script is part of the Abaqus Interface for Moldflow installation and is typically found in the *moldflow_install_dir*`/Plastic Insight 6.0/data/commands` directory.

1.3 What is the general procedure for using the Abaqus Interface for Moldflow?

The following procedure summarizes the typical usage of the Abaqus Interface for Moldflow. The remaining sections of this manual discuss these steps in detail.

To use the Abaqus Interface for Moldflow:

1. Run a Moldflow simulation.
2. Export the data as follows:
   - For a midplane Moldflow simulation, export the finite element mesh data to a file named *job-name*`.pat` and the results data (material properties and residual stresses) to a file named *job-name*`.osp`.
   - For a three-dimensional solid Moldflow simulation using Moldflow Version MPI 6, run the Visual Basic script `mpi2abq.vbs` to export the finite element mesh data to a file named *job-name*`_mesh.inp` and the results data to `.xml` files.
3. Run the Abaqus Interface for Moldflow to create a partial Abaqus input file from the Moldflow interface files.
4. Edit the Abaqus input file to add appropriate data for the analysis (for example, add boundary conditions and step data).
5. Submit the Abaqus input file for analysis.

2. Description of the translator

This chapter provides a short overview of the data generated by the Moldflow simulation. In addition, this chapter lists the assumptions used to translate the Moldflow data for the purposes of an Abaqus analysis and describes the resulting Abaqus files. The following topics are covered:

- "The Moldflow simulation," Section 2.1
- "The Moldflow interface files," Section 2.2
- "Assumptions used to translate the Moldflow data for midplane simulations," Section 2.3
- "Assumptions used to translate the Moldflow data for three-dimensional solid simulations," Section 2.4
- "Files created by the Abaqus Interface for Moldflow," Section 2.5

2.1 The Moldflow simulation

The Moldflow injection molding simulation of polymers can provide information on the thermo-mechanical properties and residual stresses of a part resulting from the manufacturing process. This information is written to interface files for subsequent finite element stress analysis.

All mechanical properties (including the effects of oriented fibers, if present) are calculated by Moldflow and written to the interface files as orthotropic constants at points through the thickness of the part. Models that contain oriented fibers are referred to as "filled"; models without oriented fibers are referred to as "unfilled."

The temperature of the model at the end of the analysis is taken to be uniform at the ambient temperature specified in the Moldflow analysis. Residual stresses due to cooling are also included, if requested.

For detailed information on obtaining a solution with Moldflow and preparing the interface files, please refer to the relevant Moldflow documentation.

Note: The Abaqus finite element mesh generated by the Abaqus Interface for Moldflow has the same topology as the Moldflow mesh except for an option to convert 4-node linear tetrahedra to 10-node quadratic tetrahedra for three-dimensional solid models. Therefore, when you create the mesh for the original Moldflow analysis, you should design a topology that is appropriate for both the mold-filling simulation in Moldflow and the structural analysis in Abaqus.

2.2 The Moldflow interface files

For midplane simulations, you must use Moldflow to create two interface files: *job-name*`.pat` and *job-name*`.osp`. Both files must use the same units.

For three-dimensional solid simulations using Moldflow Version MPI 6, the mesh and results files for filled and unfilled models are listed in Table 2–1.

Table 2–1 Interface files generated using the Visual Basic script for Moldflow Version MPI 6.

Data type	Filled model	Unfilled model
Finite element mesh data	*job-name*`_mesh.inp`	*job-name*`_mesh.inp`
Results data	*job-name*`_v12.xml`	*job-name*`_PoissonRatios.xml`
	job-name`_v13.xml`	
	job-name`_v23.xml`	
	job-name`_g12.xml`	*job-name*`_ShearModuli.xml`
	job-name`_g13.xml`	
	job-name`_g23.xml`	
	job-name`_ltec_1.xml`	*job-name*`_Ltecs.xml`
	job-name`_ltec_2.xml`	
	job-name`_ltec_3.xml`	
	job-name`_e11.xml`	*job-name*`_Moduli.xml`
	job-name`_e22.xml`	
	job-name`_e33.xml`	
	job-name`_initStresses.xml`	*job-name*`_initStresses.xml`
	job-name`_principalDirections.xml`	

The Moldflow interface files contain the following information:

Finite element mesh data:

- For midplane simulations the mesh data are in a Patran neutral file containing nodal coordinates, element topology, and element properties.
- For three-dimensional solid simulations the mesh data are in an Abaqus input file containing nodal coordinates, element topology, element properties, and boundary conditions sufficient to eliminate the structure's rigid body modes. Solid elements in the mesh files are always 4-node tetrahedra. The translator has an option to convert these to 10-node tetrahedra.

Material property data:

Elastic and thermal expansion coefficients for each element. For midplane simulations, these properties may be isotropic or orthotropic. For three-dimensional solid simulations of filled

models, these properties are orthotropic. For three-dimensional solid simulations of unfilled models, the data files contain orthotropic data adjusted to represent physically isotropic materials.

Residual stress data:

The Moldflow simulation calculates residual stresses in the plastic part after it has cooled in the mold. These residual stresses can be translated to initial stresses for the Abaqus structural analysis.

- For midplane simulations, a plane stress initial stress state is defined in the same directions as the material properties. The stress state in the material coordinates is defined in terms of the principal stresses (the shear stress is zero).
- For three-dimensional solid simulations, residual stresses for each element in *job-name*`_initStresses.xml` are in global coordinates. The translator transforms these coordinates to the same directions as the material properties.

2.3 Assumptions used to translate the Moldflow data for midplane simulations

For midplane simulations the Abaqus Interface for Moldflow translator makes a number of assumptions regarding the topology and properties of the data. These assumptions, listed below, ensure compatibility with the options available in the current release of Abaqus.

- The Moldflow mesh can consist of 3-node, planar, triangular elements as well as 2-node, one-dimensional elements that represent components such as runners and ribs. The Abaqus Interface for Moldflow translates the triangular elements to an identical mesh of Abaqus S3R shell elements. One-dimensional elements in the Moldflow mesh are not translated.
- The number of layers in the Abaqus S3R shell elements created by the Abaqus Interface for Moldflow is equal to the number of layers passed by Moldflow, which is 20. As a result, the mechanical properties and stress data passed to the Abaqus Interface for Moldflow apply to 20 layers through the thickness.
- The Abaqus input data created by the Abaqus Interface for Moldflow depend on the kind of material defined in the interface (`.osp`) file as follows:

  For unfilled isotropic materials Abaqus assumes the following:

  – A homogeneous shell formulation.

  – Isotropic material constants.

  – Abaqus section point initial stresses are interpolated from the values at the Moldflow through-thickness integration points.

  For unfilled transversely isotropic materials Abaqus assumes the following:

  – A homogeneous shell formulation.

- Transversely isotropic material constants defined for the section in terms of material principal directions plus the orientation with respect to the local Abaqus coordinate system.
- Abaqus section point initial stresses are interpolated from the values at the Moldflow through-thickness integration points.

For fiber-filled materials Abaqus assumes the following:

- A composite shell formulation.
- Lamina material constants defined for each layer in terms of material principal directions plus the orientation with respect to the local Abaqus coordinate system for each layer.
- Moldflow through-thickness integration points are taken as the midpoint of each Abaqus layer.
- Material properties are constant for each layer.
- Abaqus section point initial stresses are the same as the values at the Moldflow through-thickness integration points and constant through each layer.

The Abaqus input file that the Abaqus Interface for Moldflow generates does not contain boundary condition and load data. You must add this information to the input file manually.

2.4 Assumptions used to translate the Moldflow data for three-dimensional solid simulations

For three-dimensional solid simulations the Abaqus Interface for Moldflow translator makes a number of assumptions regarding the topology and properties of the data. These assumptions, listed below, ensure compatibility with the options available in the current release of Abaqus.

- The Abaqus Interface for Moldflow translates the tetrahedral elements to an identical mesh of Abaqus C3D4 or C3D10 solid elements (see "Execution procedure for the Abaqus Interface for Moldflow," Section 3.1, for more information).
- Orthotropic material constants are in terms of material principal directions.
- Material properties are constant for each element.
- Orientations are defined in *job-name*`_principalDirections.xml` by giving vectors defining the local 1- and 2-directions.
- Residual stresses computed by the WARP3D module of Moldflow in *job-name*`_initStresses.xml` are transformed from global coordinates to local material directions and used as initial stresses in Abaqus.
- Loads and boundary conditions representing service loads must be added to the input file manually. For simulations using Moldflow Version MPI 6, the Abaqus input file created by the translator contains boundary conditions sufficient to remove rigid body modes from the model so that an analysis to solve for the response due to initial stresses can be performed easily.

2.5 Files created by the Abaqus Interface for Moldflow

The Abaqus Interface for Moldflow reads the Moldflow interface files and creates the relevant files. The files created depend on which options you include on the command line when executing the Abaqus Interface for Moldflow. The files are described in the following sections:

- "Files created for a midplane simulation," Section 2.5.1
- "Files created for a three-dimensional solid simulation," Section 2.5.2

2.5.1 Files created for a midplane simulation

For a midplane simulation the Abaqus Interface for Moldflow creates the following three files:

Partial Abaqus input (`.inp`) file

The partial Abaqus input file contains model data consisting of nodal coordinates, element topology, and section definitions. It also contains a *STATIC step with default output requests. If you are working with isotropic materials, the input file contains material property data. Each input file begins with a series of comments that summarize the data provided by the Moldflow interface files and how the data are translated to the Abaqus input file. Additional data, such as boundary conditions and loads, and nondefault output requests must be added to this file manually.

Neutral (`.shf`) file containing material data for layered, spatially varying material properties

Material data are translated into an appropriately formatted ASCII neutral file. This file contains lamina material property data for each layer of each element. The Abaqus keywords *ELASTIC, TYPE=SHORT FIBER and *EXPANSION, TYPE=SHORT FIBER in the Abaqus input (**`.inp`**) file direct Abaqus/Standard to read material data from this file during the initialization step.

Data lines in the neutral (`.shf`) file:

First line:

1. Number of elements in the **`.shf`** file.
2. Number of layers in each shell section.

Subsequent lines:

1. Element label.
2. Layer identifier.
3. E_1.
4. E_2.
5. ν_{12}.

6. G_{12}.
7. G_{13}.
8. G_{23}.
9. α_1.
10. α_2.
11. Fiber orientation angle (in degrees), measured relative to the default element orientation.

This data line is repeated as often as necessary to define the above parameters for different layers of a shell section within different elements.

Initial stress (`.str`) file

Residual stress data from the Moldflow analysis are translated into an appropriately formatted ASCII neutral file. These data are defined in terms of the local Abaqus coordinate system at each section point. The Abaqus keyword *INITIAL CONDITIONS, TYPE=STRESS, SECTION POINTS in the Abaqus input (**`.inp`**) file directs Abaqus/Standard to read initial stress data from this file during the initialization step.

2.5.2 Files created for a three-dimensional solid simulation

If you are using an unfilled model, the Abaqus Interface for Moldflow creates only the partial Abaqus input file described below. For a three-dimensional solid simulation using a filled model, the Abaqus Interface for Moldflow may create additional files, as described below:

Partial Abaqus input (`.inp`) file

The partial Abaqus input file contains model data consisting of nodal coordinates, element topology, and section definitions. Additional data, such as service loads and boundary conditions, and nondefault output requests must be added to this file manually.

Boundary condition data sufficient to remove rigid body modes are also included.

Material (`.mpt`) file containing orthotropic material properties data

Material data from the Moldflow analysis are collected and placed in a binary file. The data written to the file are in the same form as the information provided for the Abaqus keyword *ELASTIC, TYPE=ENGINEERING CONSTANTS. These are defined in terms of the local Abaqus coordinate system of each element.

Orientation (`.opt`) file containing element orientation data

Orientations defining the directions for material properties and initial stresses are computed and placed in this binary file.

Thermal expansion (`.tpt`) file containing element thermal expansion coefficient data

The orthotropic thermal expansion data from the Moldflow analysis are collected and placed in a binary file. These are defined in terms of the local Abaqus coordinate system of each element.

3. Using the Abaqus Interface for Moldflow

This chapter describes the procedure used to create the Abaqus input files from the Moldflow interface files and how to prepare the input files for analysis. The following topics are covered:

- "Execution procedure for the Abaqus Interface for Moldflow," Section 3.1
- "Preparing the Abaqus input (`.inp`) file for analysis," Section 3.2

3.1 Execution procedure for the Abaqus Interface for Moldflow

Upon execution, the Abaqus Interface for Moldflow reads the Moldflow interface files and creates the relevant Abaqus files. The files created depend on the options included on the command line. You execute the Abaqus Interface for Moldflow using the following command:

abaqus moldflow **job**=*job-name*
[**input**=*input-name*]
[**midplane** | **3D**]
[**element_order**={**1** | **2**}]
[**initial_stress**={**on** | **off**}]
[**material**=**traditional**]
[**orientation**=**traditional**]

You can include the following options on the command line:

job

This option specifies the name of the Abaqus input files to be created. It is also the default name of the files containing the Moldflow interface data.

input

This option is used to specify the name of the files containing the Moldflow interface data if it is different from *job-name*.

midplane

This option is used to translate the results of a midplane simulation to an Abaqus model with three-dimensional (shell) elements.

3D

This option is used to translate the results of a three-dimensional solid simulation to an Abaqus model with solid elements.

element_order

This option is used to specify the order of elements created in the partial input file for three-dimensional solid simulations. Possible values are **1** to create first-order elements (C3D4) and **2** to create second-order elements (C3D10). The default value is **2**. This option is valid only when using the **3D** option.

initial_stress

This option specifies whether or not initial stress will be included in the model. This option is valid only when using the **3D** option.

If the **initial_stress** option is not included or **initial_stress=off**, initial stresses will not be translated.

If **initial_stress=on**, initial stresses will be written to the input file.

material

This option is used to specify where the material properties are written. If **material=traditional**, the material properties will be written to the input file. Otherwise, the material properties will be written to the (binary) **.mpt** file. Using **material=traditional** is not recommended for large models for performance reasons, since every element will have its own *MATERIAL definition.

orientation

This option is used to specify where the orientations are written. If **orientation=traditional**, the orientations are written to the input file. Otherwise, the orientations will be written to the (binary) **.opt** file. Using **orientation=traditional** is not recommended for large models for performance reasons, since every element will have its own *ORIENTATION definition.

3.2 Preparing the Abaqus input (.inp) file for analysis

Once the Abaqus Interface for Moldflow has created the Abaqus input (**.inp**) file, you must complete the input file manually before submitting it for analysis. Refer to the Abaqus Analysis User's Manual for detailed information on performing an Abaqus analysis.

3.2.1 Preparing for a shrinkage and warpage analysis

A shrinkage and warpage analysis calculates the deformation caused by the residual stresses in the model after it is removed from the mold. Usually only rigid body modes must be removed.

In this case you must ensure that residual stresses have been translated. For three-dimensional solid Moldflow simulations boundary conditions sufficient to restrain rigid body modes are automatically

translated to the input file. In other cases you are required to add appropriate boundary conditions to remove the rigid body modes of the model.

In certain cases problems with convergence can occur when you must account for geometric nonlinearity and large initial stresses are present. You can overcome these problems by using two analysis steps:

- In the first step constrain all displacement degrees of freedom.
- In the second step use the OP=NEW parameter to apply boundary conditions that remove the rigid body modes.

3.2.2 Preparing for a service loading analysis

A service loading analysis (with appropriate boundary conditions) assesses the performance of the model. You can perform this analysis with or without initial stresses. You must specify the appropriate boundary conditions and loads as history data in the Abaqus input file.

3.2.3 Preparing for other analysis types

Any Abaqus/Standard analysis procedure can be performed with the translated model provided that you specify the correct boundary conditions and loading in the Abaqus input file. In addition, certain analysis types may require you to specify additional material constants, model data, and/or solution controls in the input file.

4. Examples

This chapter provides examples of Moldflow models translated to Abaqus input files by the Abaqus Interface for Moldflow. All of the files associated with these examples are included with the Abaqus release. The following topics are covered:

- "Extracting example problem files," Section 4.1
- "Example 1: Natural frequency analysis of a fiber-filled bracket," Section 4.2
- "Example 2: Natural frequency analysis of an unfilled bracket," Section 4.3
- "Example 3: Deformation due to initial stresses in a three-dimensional filled bracket," Section 4.4

4.1 Extracting example problem files

You can use the Abaqus **fetch** utility to extract example problem files from the compressed archive files provided with the Abaqus release. To extract all of the relevant files for a particular example problem, you must enter the following commands:

Example 1: Natural frequency analysis of a fiber-filled bracket

```
abaqus fetch job=moldflow_ex1*
```

Example 2: Natural frequency analysis of an unfilled bracket

```
abaqus fetch job=moldflow_ex2*
```

Example 3: Deformation due to initial stresses in a three-dimensional filled bracket translated from Moldflow Version MPI 6

```
abaqus fetch job=bracket3d_mpi6*
```

For more information on using wildcard expressions with the Abaqus **fetch** utility, see "Fetching sample input files," Section 3.2.14 of the Abaqus Analysis User's Manual.

4.2 Example 1: Natural frequency analysis of a fiber-filled bracket

The bracket in this example consists of 926 nodes and 1719 S3R elements. The model contains seven different element sets. Each element set has a different thickness and is modeled as a laminated composite with 20 layers.

Ten unrestrained vibration modes are computed. The first six frequencies are approximately zero. The frequencies for the first four flexible modes are listed in Table 4–1.

Table 4–1 Frequencies for the first four flexible modes for the fiber-filled bracket.

Mode	Frequency, Hz
7	334
8	430
9	740
10	752

The Abaqus finite element model is shown in Figure 4–1.

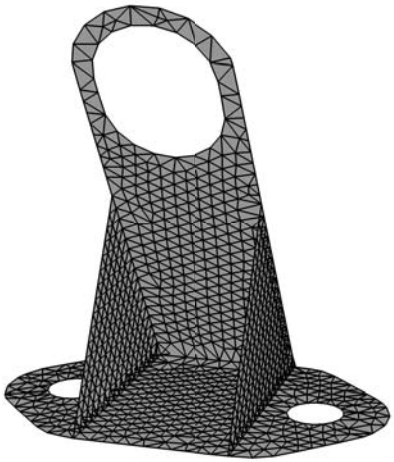

Figure 4–1 Finite element mesh of the fiber-filled bracket.

The complete input file, **`moldflow_ex1.inp`**, is shown below.

```
**************************************************************
**
** translated data from the Moldflow interface files named
```

```
** "moldflow_ex1.pat"
**  and
** "moldflow_ex1.osp"
**
** to the following Abaqus input files:
**
**            input file = moldflow_ex1.inp:  YES
**
** neutral material file = moldflow_ex1.shf:  YES
**                         (for *ELASTIC/*EXPANSION data)
**
**   initial stress file = moldflow_ex1.str:  YES
**                         (for *INITIAL CONDITIONS data)
**-------------------------------------------------------------
** echo of header information from the Moldflow interface
** files:
**
** TITL information from .osp file:
** TITL
**
** FILE information from .osp file:
** FILE  JUN14-2002  11:13:29 mpi310 Residual Stress &
** Properties
**
**      number of nodal properties = 0
**    number of element properties = 13
**                  number of nodes = 926
**           number of TRI elements = 1719
**            number of 1D elements = 0
**
** Moldflow results were written with ISP coding, i.e.,
** this is a filled anisotropic material with residual stresses
**           ------ -----------          ----
**-------------------------------------------------------------
** this input file was created with the following keyword data:
**
**      *NODE     (926 nodes)
**
**      *ELEMENT  (1719 S3R elements)
**
**      *SHELL SECTION, COMPOSITE  (7 ELSETs)
```

```
**
**      *MATERIAL
**      *ELASTIC, TYPE=SHORT FIBER
**      *EXPANSION, TYPE=SHORT FIBER
**       (elastic and expansion data will be read from file
**        moldflow_ex1.shf)
**
**      *INITIAL CONDITIONS, TYPE=STRESS, SECTION POINTS,
**       INPUT=moldflow_ex1.str
**
**      *STEP
**       (Dummy step data.  Loads and boundary conditions
**        may need to be added to complete the model.)
**********************************************************************
*HEADING
TITL
**********************************************************************
*NODE, NSET=NALL, INPUT=moldflow_ex1_nodes.inp
**********************************************************************
*ELEMENT, TYPE=S3R, ELSET=EALL, INPUT=moldflow_ex1_elements.inp
**********************************************************************
*INCLUDE, INPUT=moldflow_ex1_elsets.inp
**********************************************************************
*INCLUDE, INPUT=moldflow_ex1_sections.inp
**********************************************************************
*MATERIAL,NAME=moldflow_mat_01
*ELASTIC,TYPE=SHORT FIBER
*EXPANSION,TYPE=SHORT FIBER
*DENSITY
1500.,
**********************************************************************
*INITIAL CONDITIONS,TYPE=STRESS,SECTION POINTS,
 INPUT=moldflow_ex1.str
**********************************************************************
*STEP
*FREQUENCY, EIGENSOLVER=LANCZOS
10,
*END STEP
**
** SIGN: ljoXzAUD!NPKmw==
**
```

4.3 Example 2: Natural frequency analysis of an unfilled bracket

This example uses the same Abaqus finite element model as Example 1, but the material properties are transversely isotropic. The shell section definition is homogeneous instead of composite. Twenty-one equally spaced Simpson integration points are used through the shell thickness.

The frequencies for the first four flexible vibration modes of the unfilled bracket are listed in Table 4–2. The unfilled material in this example is softer than the filled material in Example 1 and, consequently, the frequencies are lower.

Table 4–2 Frequencies for the first four flexible modes for the unfilled bracket.

Mode	Frequency, Hz
7	146
8	217
9	363
10	371

The complete input file, **moldflow_ex2.inp**, is shown below.

```
**************************************************************
**
** translated data from the Moldflow interface files named
** "moldflow_ex2.pat"
**  and
** "moldflow_ex2.osp"
**
** to the following Abaqus input files:
**
**            input file = moldflow_ex2.inp:  YES
**
** neutral material file = moldflow_ex2.shf:  NO
**                         (for *ELASTIC/*EXPANSION data)
**
**   initial stress file = moldflow_ex2.str:  YES
**                         (for *INITIAL CONDITIONS data)
**------------------------------------------------------------
```

```
** echo of header information from the Moldflow interface
** files:
**
** TITL information from .osp file:
** TITL
**
** FILE information from .osp file:
** FILE  JUN14-2002  11:19:59 mpi310 Residual Stress &
** Properties
**
**      number of nodal properties = 0
**    number of element properties = 45
**                  number of nodes = 958
**           number of TRI elements = 1719
**            number of 1D elements = 32
**
** Moldflow results were written with IST coding, i.e.,
** this is an unfilled anisotropic material with residual
** stresses
**            -------- -----------          ----
**--------------------------------------------------------------
** this input file was created with the following keyword data:
**
**      *NODE     (926 nodes)
**
**      *ELEMENT  (1719 S3R elements)
**
**      *SHELL SECTION, COMPOSITE  (7 ELSETs)
**
**      *MATERIAL
**      *ELASTIC, TYPE=SHORT FIBER
**      *EXPANSION, TYPE=SHORT FIBER
**       (elastic and expansion data will be read from file
**        moldflow_ex2.shf)
**
**      *INITIAL CONDITIONS, TYPE=STRESS, SECTION POINTS,
**       INPUT=moldflow_ex2.str
**
**      *STEP
**       (Dummy step data.  Loads and boundary conditions
**        may need to be added to complete the model.)
```

```
**************************************************************
*HEADING
TITL
**************************************************************
*NODE, NSET=NALL, INPUT=moldflow_ex2_nodes.inp
**************************************************************
*ELEMENT, TYPE=S3R, ELSET=EALL, INPUT=moldflow_ex2_elements.inp
**************************************************************
*INCLUDE, INPUT=moldflow_ex2_elsets.inp
**************************************************************
*INCLUDE, INPUT=moldflow_ex2_sections.inp
**************************************************************
*MATERIAL,NAME=moldflow_mat_01
*ELASTIC,TYPE=SHORT FIBER
*EXPANSION,TYPE=SHORT FIBER
*DENSITY
1500.,
**************************************************************
*INITIAL CONDITIONS,TYPE=STRESS,SECTION POINTS,
 INPUT=moldflow_ex2.str
**************************************************************
*STEP
*FREQUENCY, EIGENSOLVER=LANCZOS
10,
*END STEP
```

4.4 Example 3: Deformation due to initial stresses in a three-dimensional filled bracket

This example uses a solid Abaqus finite element model that is similar to the model used in Example 1.

To execute the Abaqus Interface for Moldflow, enter the following command:

```
abaqus moldflow job=bracket3d_mpi6 3D initial_stress=on
```

A contour plot of initial stresses is shown in Figure 4–2.

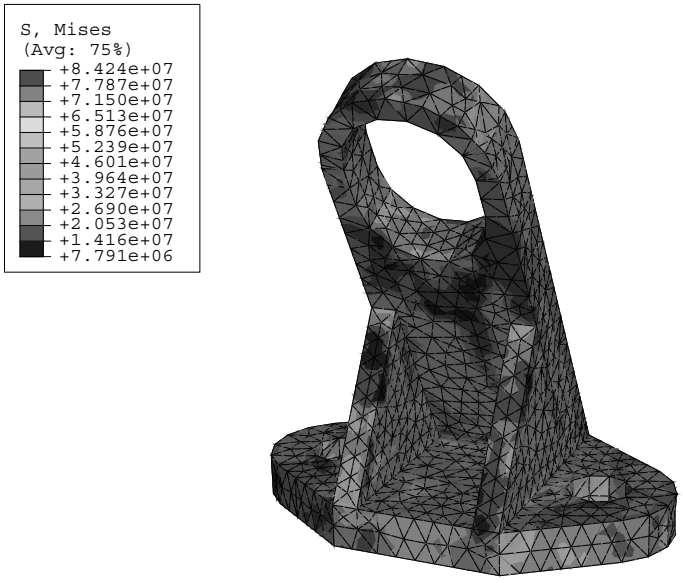

Figure 4–2 Contour plot of the initial stresses for the filled bracket using Moldflow Version MPI 6.

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis; multiphysics solutions for insight into challenging engineering problems; and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Vélizy, France, SIMULIA provides sales, services, and support through a global network of regional offices and distributors. For more information, visit www.simulia.com.

About Dassault Systèmes

As a world leader in 3D and Product Lifecycle Management (PLM) solutions, Dassault Systèmes brings value to more than 100,000 customers in 80 countries. A pioneer in the 3D software market since 1981, Dassault Systèmes develops and markets PLM application software and services that support industrial processes and provide a 3D vision of the entire lifecycle of products from conception to maintenance to recycling. The Dassault Systèmes portfolio consists of CATIA for designing the virtual product, SolidWorks for 3D mechanical design, DELMIA for virtual production, SIMULIA for virtual testing, ENOVIA for global collaborative lifecycle management, and 3DVIA for online 3D lifelike experiences. Dassault Systèmes' shares are listed on Euronext Paris (#13065, DSY.PA), and Dassault Systèmes' ADRs may be traded on the US Over-The-Counter (OTC) market (DASTY). For more information, visit www.3ds.com.

www.3ds.com Abaqus, the 3DS logo, SIMULIA, CATIA, SolidWorks, DELMIA, ENOVIA, 3DVIA, and Unified FEA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries. Other company, product, and service names may be trademarks or service marks of their respective owners. © Dassault Systèmes, 2012

DASSAULT SYSTEMES